Injector Spray Combustion Modeling

T.S. Wang/ED32
205–544–0503
E-mail: Ten-see.Wang@msfc.nasa.gov

Injector design and combustion chamber compatibility are among the key technologies in developing a new generation of cost effective high thrust-to-weight ratio propulsion systems for the reusable launch vehicles. A design analysis tool for predicting the complex combustion flowfield near the injector plate can be very useful for the engine design team in enhancing the engine performance leading to increased thrust-to-weight ratio.

The main objectives of this project is to synthesize relevant state-of-the-art physical sub-models into an advanced computational fluid dynamics (CFD) code for modeling spray combustion using proven methodologies and to validate the related numerical and physical models systematically against available experimental data from the open literature and other ongoing test programs. The physical models include turbulence/droplet interaction, dense spray atomization, interphase transport, chemical reaction, droplet sub/super-critical vaporization and combustion. A state-of-the-art volume-of-fluid (VOF) method for spray atomization and combustion is developed. Droplet atomization/breakup mechanism along the gas-liquid interfaces is modeled by empirical correlations applicable to coaxial injector elements. A robust point implicit finite-rate chemistry module is also employed to model the gas-phase chemical reactions.

Benchmark validation test cases for water spray and hot-flow spray combustion flows were investigated to validate the present numerical model. Good results were obtained in the validation processes. Applications of the present technology to the liquid rocket engine injection systems have also shown promising results. It is expected that the presented technology will be developed into a user-friendly design tool to be incorporated in the liquid rocket engine industry and many other commercial and government industrial applications.

In summary, a computational spray combustion technology has been developed for liquid rocket engine chamber flow analysis. An injector spray combustion design tool will be developed into a commercial product which will allow the present technology to be widely used by the industry.

Sponsor: Engineering Science, Inc., Huntsville, AL

Biographical Sketch: Dr. Ten-See Wang is currently a team leader of the Computational Analysis Team in the Fluid Dynamics Branch. He received his Ph.D. from Louisiana State University in 1980. He has previously been affiliated with SAIC, Continuum, SRA, and SECA, Inc. His recent work has been focusing on applying CFD methods for propulsion system and launch vehicle environments.